Finite Element Analysis of Lime Stone Slurry Tank

Vinayak M. Patil[1], Prof. M.M. Joshi[2]

[1]*Student of M.E. (Design Engineering), NBN SSoE, Ambegao -Pune, Savitribai Phule Pune University, India*

[2]*Faculty member of Mechanical Engineering, NBN SSoE, Ambegao -Pune, Savitribai Phule Pune University, India*

***Abstract*—Limestone slurry is highly effective wet scrub to reduce toxic emissions because it is cheap plentiful, limestone is the most commonly used reagent for this purpose.**

The slurry reacts well with the toxic sulfur dioxide and makes the waste less hazardous to the environment. The waste then may be dewatered and deposited safely in landfills or sold as an ingredient for the manufacturing of gypsum wallboard and cement. It may even be used as a fertilizer additive. Limestone stays only slightly soluble in water. The slurry needs regular agitation, otherwise, the suspended particles soon settle out and form a solid.

A limestone slurry tank designed with the objective of keeping the slurry agitated for storage purpose. Besides structural stability, the internal pressure indicates the risk of leakage and failure of slurry tank, hence FEA is used to identify the critical zone and hence be manipulated it through a structural modification to make the system safe.

Keywords: **Design, fem, surface mesh, analysis, optimization, ANSYS, skirt support.**

I. INTRODUCTION

A limestone slurry tank designed with the objective of keeping the slurry agitated for storage purpose. Besides structural stability, the presence agitator indicates risk of vibration failure too, hence FEA is used to identify natural frequency of tank, and hence be manipulated it through a structural modification to make the system safe. Limestone slurry is sprayed into the sulfur dioxide containing gas stream. The chemical reaction in the recalculating limestone slurry and reaction products may be carefully controlled in order to maintain the desired sulfur dioxide removal efficiency and to prevent operating problems. Though the process is highly efficient, it depends on lime stone being in wet slurry form.

However limestone is just slightly soluble in water, and tendency of settling down into solid particles. These solid particles can then obstruct flow and damage the equipment. Hence it is necessary to maintain continuous agitation for slurry to prevent deposition of lime.

II. LITERATURE REVIEW

Jaroslave Mackerle, linkoping institute of technology in“finite elements in the analysis of pressure vessel and piping” gives the review of the different possibilities that exist today for the finite element analysis in the fields of pressure vessels and piping.

J.L. Gonzalez,S. Gomez.“Analysis of mechanical behavior of a delayed coker drum with circumferentially cracked skirt” discussed the reasons for the failure of skirt suupot, analyzed them and optimized the design.

Michael A. Porter and Dennis H. Martens in “On Using Finite Element Analysis for Pressure Vessel Design” review of the use of PC-based Finite Element software in the analysis of typical pressure vessel components.

Busuioceanu (Grigorie) Paraschivaa, Stefanescu Mariana-Florentinab, Ghencea Adrianc in there “Study of Stresses and Stress Concentrations in Pressure Vessels” aims to present an original methodology based on the theory of unitary bending moments, characteristic for shells of revolution, respectively shorter structural theory.they study the different type of stresses such as stresses due to standard gravity ,pressure on various parts of the pressure vessel.

A. Kazemi Amiri and C.Bucher Compares the experimental results with the numerical simulation, where actual loads are available and and analysis results of wind load calculation which are in nearby agreement.

Kadhim Hussein Mukhirmesh in “Design and analysis of welded joints of pressure vessel” used the Mathematical correlations for the design of pressure vessel whose design parameters are specified by a company according to the required weld efficiency. Modeling is done in Pro/Engineer. He did the Structural analysis in Ansys on the welded joint of pressure vessel for different weld efficiencies.

Dennis H. Martens in “On Using Finite Element Analysis for Pressure Vessel Design” review of the use of PC-based Finite Element software in the analysis of typical pressure vessel components.

Arter Calnins in “stress classification in pressure vessel and piping” takes the basic concept of primary and secondary stresses and explained the limits.

Manish M. Utagikar and Prof. S.B. Naik in “Finite

Element Analysis of Elliptical Pressure Vessels" designed the model of pressure vessel with elliptical head and performed the analysis.

Objective and Methodology

All the analyses were performed by means of FEM, using the ANSYS finite element program. The finite element method (FEM) is a numerical technique for finding approximate solutions of partial differential equations (PDE) of physics and engineering by discretization of the domain of analysis into elements.

Error Analysis in FEA helps us to rationalize FEA computation, understand element performance and pathological problems, and also provides a sound basis for developing new elements.

The work presented here focuses on determining and analyzing the different stresses developed in pressure vessel due to internal pressure and wind load and to optimize the design for safety and economy.

Material Properties:

This steel was already been used for the manufacture of many pressure vessels. Although practical experience in welding and PWHT are not really a problem with this steel. Where in order to avoid brittle fracture, the simple method exposed is limited to steels with maximum yield strength. The SA 516 Grade 70 fracture toughness evolution with temperature was obtained from the results of Charpy impact tests performed on welded specimens with the notch placed in the heat affected zone at 1 mm from the fusion line (coarse grain region). Material used for the pressure vessel is SA516 GR70 Steel. The properties of the material are described in following table.by using Reference of ASME Code Section-VIII, Div-I.

Material Specification

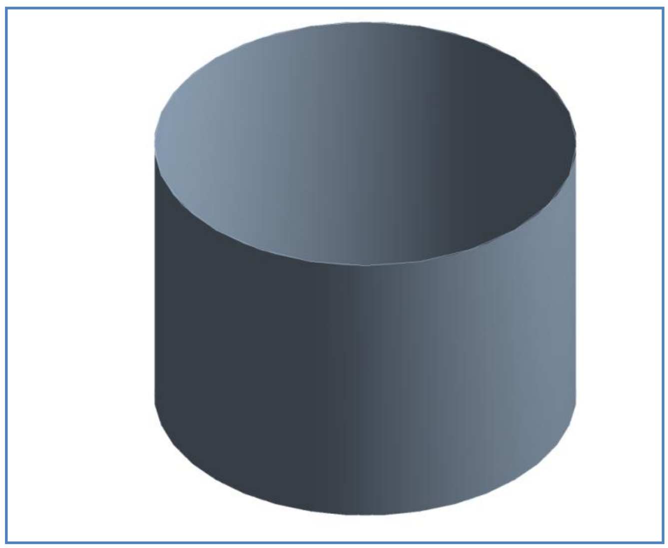
Sr.no.	Description	Quantity
1	Yield Strength	139 MPa
2	Ultimate strength	354 MPa
4	Modulus of Elasticity	200 GPa
5	Thermal conductivity	60.5 W/mk

Given data and calculation result:

Internal Pressure	P	0.32 MPa
External Pressure	P0	Atmospheric
Process Volume	Vp	282 m^3
Expected Stagnant Volume	Vs	5.5 m^3
Buffer Volume Requirement	Vb	3.2 m^3
Radius Of Vessel	R	3000 mm
Total Length	H	14000 mm
Total no.of nozzle		4
Inlet Nozzle Diameter	Dn	1000 mm
Two outlet Nozzle Diameter	D_o	500 mm
Drain nozzle Diameter		1000 mm
Support Height		4000 mm
Head Type		Ellipsoidal Head
Thickness of shell		12 mm
Thickness of nozzle wall		12 mm
Thickness of flange		12 mm
Initial thickness of RF pad		12 mm

A) Design of Skirt Support

Design of Skirt support for vertical pressure vessel, kept wind load of velocity 41 m/s, with the height of 4m, standard values of Skirt directly taken from josh's process equipment design data book as per requirement of shell diameter of vessel.



Skirt support

Element Name	Thickness & Width (mm)
Thickness of Skirt	12 mm
OD of Skirt	6024 mm
Height of Skirt	4000 mm
Joint Type	Welded Joint

III. MESHING

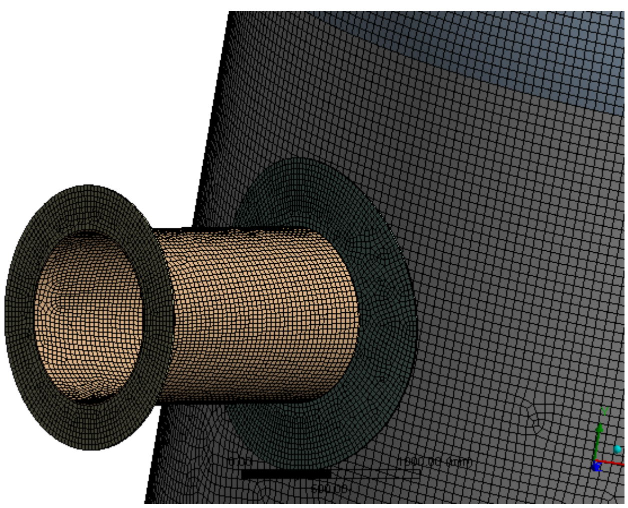
Basic methodology of finite element method is to prepare calculations at only limited number of points and then interpolate the results for entire domain i.e. surface and volume. Any continuous object has infinite number of degrees of freedom and it is impossible to solve it for required results in this format. Finite element method reduces degrees of freedom from infinite to finite with the help of Discretization of entire domain. Various types of elements like 1D, 2D, 3D, mass, spring, damper, gap etc. are available for meshing, one has to select them depending upon the geometry, size and shape of the component, type of the analysis to be carried out and time availability for completion of project.

Steps Involved in Meshing Pressure Vessel in ANSYS

- Selection of element type for meshing.
- Creating simplified parts & meshing the parts.

Obtaining desirable mesh pattern for different size and quality

Sr. No.	component	Element type	Element size	
1	Shell	Hex dominant	250	200
2	Skirt	Hex dominant	250	200
3	Dome	Hex dominant	250	200
4	Nozzle	Hex dominant	60	24
5	RF Pad	Hex dominant	30	24
6	Flange	Hex dominant	30	24
Total Nodes			233219	469788
Total Elements			46384	90368



With this much high no. nodes and no. of elements it is more time consuming, complex and laborious task of FEM analysis for this system of configuration.

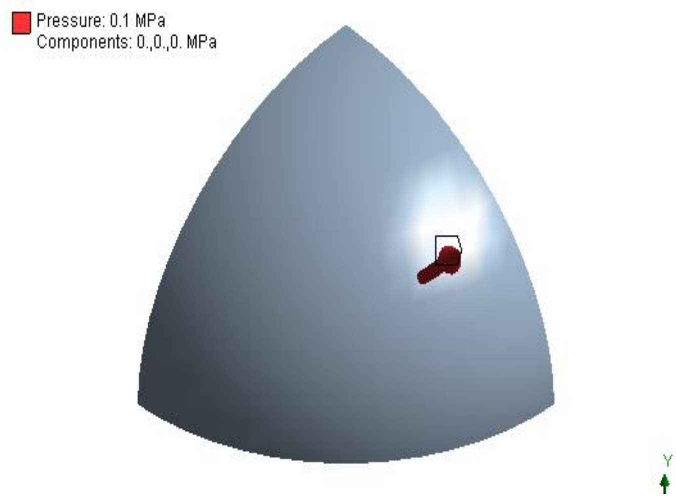
Hence surface model of pressure vessel is created using same element and element sizing.

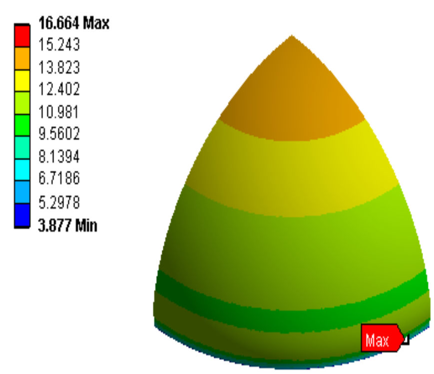
It is observed that with same element and element sizing the no. of nodes and no. of elements generated are too less. i.e.

Sr. No.	component	Element type	Element size
1	Shell	Hex dominant	200
2	Skirt	Hex dominant	200
3	Dome	Hex dominant	200
4	Nozzle	Hex dominant	24
5	RF Pad	Hex dominant	24
6	Flange	Hex dominant	24
Total Nodes			32469
Total Elements			30661

IV. SELECTION OF ELEMENT FOR MESHING

For the selection of meshing element, one component of the pressure vessel is selected for analysis. Then meshing is generated using different types of elements in sold and surface meshing for same no. of total nodes (say 100000) and results are observed.



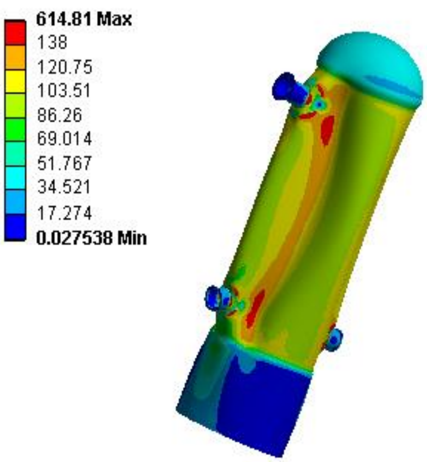


From above analysis it is concluded that the shell 96 element gives the same results as that of the solid mesh (Hexagonal and Tetrahedron) but with the very less no. of nodes and less no. of elements with small elemental size. Hence selection of shell 96 minimizes the load on the system and saves the time for analysis with the same accuracy. The element has six degrees of freedom at each node: translations in the nodal x-direction, y-direction, and z- directions and rotations about the nodal x-axes, y-axes, and z-axes. The deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities.

Hence for further analysis and optimization we use “shell 96” element.

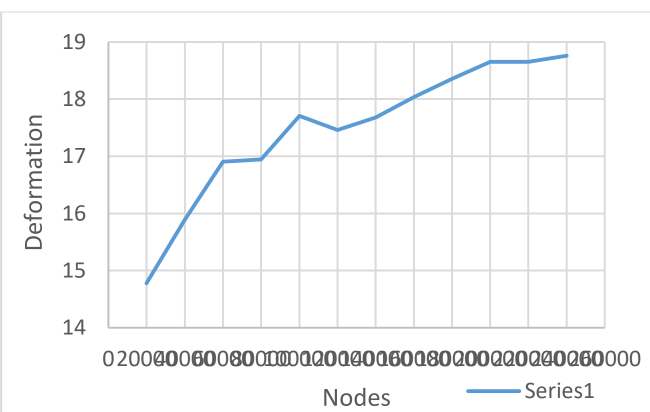
V. SURFACE MESH NODE SELECTION

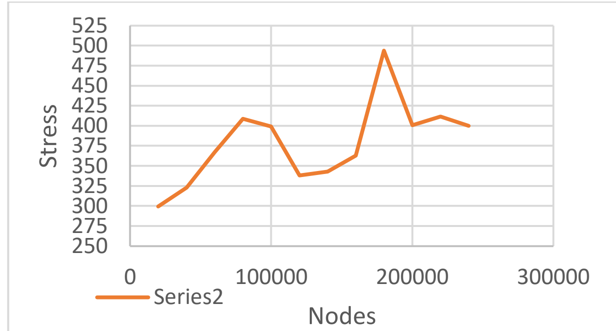
With the shell 96 element surface meshing is done. The forces like standard gravitational field and internal pressure is applied. With this applied forces and boundary conditions solution is generated for different no. of nodes. The stresses developed and total deformation for different no of nodes is done



The graph is ploted :

- Nodes Vs Total deformation
- Nodes vs equivalent stress





From above graph it is clear that maximum stress is developed at 180000 nodes. Before and after 180000 nodes stress values for different no. of nodes are less. Hence 180000 no. of nodes can be termed as critical no. of nodes.

VI. RF PAD OPTIMIZATION

When solution is generated with given boundary conditions and designed specifications it is observed that the maximum stress is developed at RF pad which is very high as compared to ultimate strength of the material used for the RF pad.

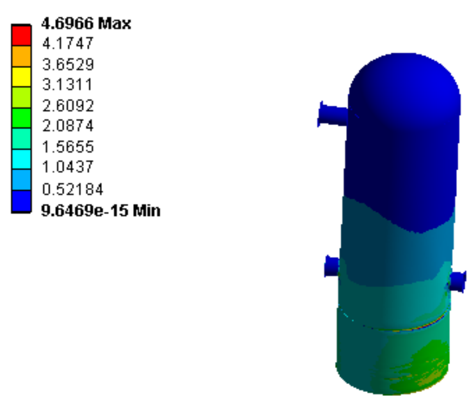
After no. of analysis for different values of RF Pad thickness analysis is done till stress developed in pressure vessel material is under control.

From analysis it is concluded that the thickness of RF Pad thickness of 16 mm the thickness developed is under control and is less than ultimate strength of material.

Hence finalized thickness of RF Pad is 15 mm.

VII. WIND LOAD ANALYSIS

As total height of pressure vessel is 14 m it is necessary to perform a wind load analysis on pressure vessel for safety reasons. By meteorological conditions and observations taking average wind speed as 41 m/s wind pressure wind load acting on pressure vessel is calculated and further analysis is performed.



Stress due to wind load

From above it is clear that pressure vessle is safe under wind load. Hence our design is safe for wind load and safety is assured.

VIII. CONCLUSION

A FEM analysis was used to understand the behavior of a pressure vessel made of high strength steel subject to the design loads and assuming the existence of the ‘‘worst case’’ crack allowed by the ASME standards in order to demonstrate the safe use of these steels and the too conservative design rules currently applied by the ASME manufacture codes.

It is imperative to use FEA tool to understand behavior and interaction of these multiple forces. Stress concentration is needed to be analyzed especially at critical due to pressure load of agitating slurry.

FEA has task of both ensuring safety and plus identifying the location for testing

REFERENCES

[1]. Jaroslave Mackerle, linkoping institute of technology,S-581 Linkoping, Swedan “ finite elements in the analysis of pressure vessel and piping” p .279-280

[2]. J.L. Gonzalez,S. Gomez, XXIV Italian group of fracture conference,1-3 march 2017,Italy.“Analysis of mechanical behavior of a delayed coker drum with circumferentially cracked skirt”.p 36-38

[3]. Michael A. Porter, Dennis H. Martens Article in American Society of Mechanical Engineers “On Using Finite Element Analysis for Pressure Vessel Design”. Pressure Vessels and Piping Division (Publication) PVP January 1999

[4]. Busuioceanu (Grigorie) Paraschivaa, Stefanescu Mariana-Florentinab, Ghencea Adrianc “ Study of Stresses and Stress Concentrations in Pressure Vessels.” Journal of Business Economics and Information Technology

[5]. A.kazemi Ameri, C. Bucher Vienna doctoral programme on water resourse system,A-1040,Vienna,Astria.”A procedure for in situ wind load reconstruction from structural response only based on field testing data.”(2017)75-86.

[6]. Kadhim Hussein Mukhirmesh*1, Dr. P. Laxminarayana Kadhim Hussein Mukhirmesh et. al./ International Journal of engineering & science research “Design and analysis of welded joints of pressure vessel” IJESR/May 2015/ Vol-5/Issue-5/341-34

[7]. Dennis H. Martens Article in American Society of Mechanical Engineers “On Using Finite Element Analysis for Pressure Vessel Design”. Pressure Vessels and Piping Division (Publication).

[8]. Arter Calnins, mechanical enng. Dept.lehigh university bathlehem “stress classification in pressure vessel and piping” PA 18015-3015.

[9]. Manish M. Utagikar, Prof. S.B. Naik “Design and analysis of welded joints of pressure vessel.” IJESR/May 2015/ Vol-5/Issue-5/341-346